CONVERTER MAGAZINE
Volume 2021, No. 7

Research on Ventilation Simulation of Underground Complex Traffic Tunnel Group Based on FMK

Zhenggang Zhan[1], Jinhai Zeng[1], Zhiqiang Hua[1,2*], Yao Liu[1], Meng Liu[3]
[1]PowerChina Guiyang Engineering Corporation Limited, Guiyang, Guizhou, China
[2]Guizhou Building Information Model (BIM) Engineering Technology Research Center, Guiyang, Guizhou, China
[3]Dassault Systèmes (Shanghai) Information Technology Co., Ltd, Shanghai, China
**Corresponding Author.*

Abstract

To facilitate the construction of underground powerhouse, dam and other areas and the operation and management of power station, a considerable quantity of underground traffic tunnels need to be built. These traffic tunnel groups are of considerable quantity, with complex trend, large bends and large slopes. The smoke concentration, harmful gas accumulation, ventilation rate and fire smoke exhaust in the tunnels are key problems to be solved in the design and operation of tunnel ventilation system of underground complex traffic tunnel groups. At the same time, the airflow direction and velocity in the tunnel are uncertain, so it is difficult to form effective airflow organization by using conventional ventilation design methods. To solve the ventilation design of large-scale complex underground traffic tunnel groups, this study simulates ventilation design airflow organization of large-scale complex underground traffic tunnel groups by FMK simulation based method to obtain ventilation characteristics and design key parameters of large-scale complex underground traffic tunnel groups, which provides reference for engineering design and practical application.

***Keywords**: Traffic tunnel group, Tunnel ventilation system, Gas flow direction, Gas flow velocity, FMK simulation*

I. Introduction

The Qinghai-Tibet Plateau in China is characterized by high mountains and deep valleys. To build large hydropower stations in this area, it is necessary to build a considerable quantity of underground complex traffic tunnels for the convenience of the construction of underground powerhouse and dam area and the operation and management of power stations. These traffic tunnels are characterized by considerable quantity, complex trend, large bends and large slopes. Due to the closed space of underground complex traffic tunnel, the smoke concentration, harmful gas accumulation, ventilation rate and fire smoke exhaust in the tunnel will be the problems to be concerned in the ventilation design and operation of underground complex traffic tunnels [1-2]; at the same time, unlike the conventional single and straight road tunnel ventilation, the underground complex traffic tunnels of large hydropower stations have considerable quantity, complex trends, large bends and large slopes, and the airflow direction and velocity in the tunnels are uncertain, so it is difficult to form effective airflow organization by using conventional ventilation design methods [3-5].

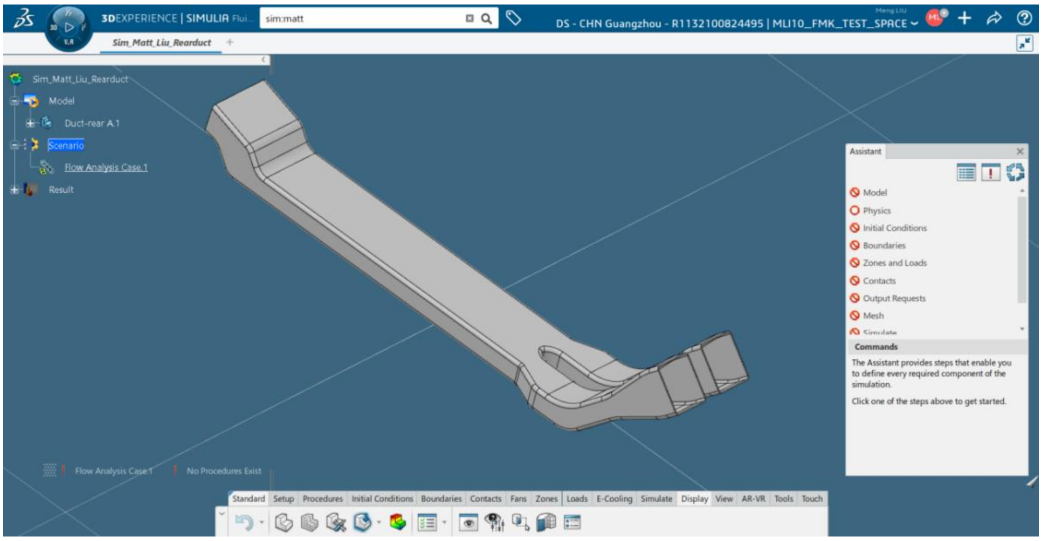
Many scholars have studied tunnel ventilation in numerous aspects. For example, three-dimensional turbulent RNG k-e turbulence model is used to analyze the ventilation characteristics and fire smoke characteristics in the construction of deep-buried long tunnel of water diversion project. The ventilation situation of deep-buried long tunnel of water diversion project under different working conditions is numerically simulated, and the final numerical equation of ventilation characteristics is obtained to realize numerical simulation [6]; some studies have studied the feasibility of adopting double-hole complementary ventilation system in spiral tunnel by theoretical design and numerical simulation, and concluded that when adopting double-hole complementary ventilation system, the effect of local concentration of pollutants in tunnel can be weakened, and the local concentration of air pollutants in tunnel can be prevented from exceeding the standard [7-8]; in other studies, hydrodynamics analysis model is adopted, and CO is used as tracer pollutant. The influence of temperature difference between inside and outside tunnel, airflow velocity in canyon and airflow velocity ratio between inlet and outlet on pollutant crossflow is analyzed, and a green energy-saving ventilation scheme is put forward [9]. In these studies, the ventilation characteristics of various tunnels are analyzed by means of simulation and experimental study [10-12], but most of the studies are mainly aimed at single tunnels or straight tunnels, and do not involve the analysis of ventilation characteristics when multiple intersecting tunnels form complex tunnel groups [13-15].

To solve the ventilation design of large-scale underground complex traffic tunnel groups, this paper will use FMK simulation to simulate the ventilation of large-scale underground complex traffic tunnels to obtain the ventilation characteristics and key design parameters of large-scale underground complex traffic tunnels, which can provide reference for engineering design.

II. Calculation Method

2.1 Introduction of software

FMK (Fluid Dynamics Engineer) is a CFD computing tool developed by Dassault, France based on 3D Experience platform. The simulation field is introduced into 3D Experience, which innovates the use scenarios and working methods of CFD tools and provides simulation roles for engineers who perform/perform routine fluid flow and heat calculations. FMK simulates the characteristics of fully guided working mode and fully automatic close-fitting engagement, so that non-fluid experts can set up simulation scenarios in a few minutes and get fluid analysis results in the design stage. It minimizes user interaction, significantly improves the turnover efficiency of design iteration, and truly realizes "out of the box". The operation interface is shown in Figure 1.



The Assistant provides steps that enable you to define every required component of the simulation.

Click one of the steps above to get started.

Fig 1: FMK operation interface

The computer theory of FMK simulation is an industry standard RANS solver which adopts high-quality mixed grid, grid tolerance near-wall processing and thorough verification. It can further reduce the turnaround time of accurate fluid flow and heat transfer simulation. Integrating CAD+CFD+PLM and guided user experience, it provides customized design schemes for engineers, and efficiently implements product design and corresponding modification.

2.2 Software principle

FMK conducts ventilation simulation calculation based on 3D3D Experience platform, and its calculation flow is shown in figure 2.

Fig 2: Flow chart of FMK algorithm

FMK simulation takes Reynolds equation as the basic governing equation, and establishes RANS Realizable K-e turbulence model to close the equation and realize stable turbulence calculation and long-term transient simulation [16], as shown in Figure 3

$$v_i(x,t) = V_i(x) + v_i'(x,t)$$

Average speed ↑ ↑ **Pulsating velocity**

$$V_i(x) = \lim_{x\to\infty}\frac{1}{\mathrm{T}}\int_{\mathrm{t}}^{\mathrm{t+T}} v_i(x,t)\mathrm{dt},\quad \overline{v_i'}(x,t)=0,\quad \frac{\partial V_i}{\partial x_i}=0$$

$$\rho\frac{\partial V_i}{\partial t}+\rho \mathrm{V_j}\frac{\partial V_i}{\partial x_\mathrm{j}} = -\frac{\partial \mathrm{P}_i}{\partial x_i}+\frac{\partial}{\partial x_\mathrm{j}}(\mu(\frac{\partial V_i}{\partial x_j}+\frac{\partial V_j}{\partial x_i})-\overline{\rho v_j'v_x'})$$

$$\tau_{ij} = -\overline{\rho v_i'v_j'} \leftarrow \textbf{Reynolds stress tensor}$$

Fig 3: Reynolds control equation

The internal and external fluid flows are analyzed by an automatically selected turbulence model (mixed wall function). The mixed wall function consists of wall function and near wall function. When the boundary layer grid is thick, the wall function is used for processing; when the boundary layer grid is dense, the calculation efficiency and accuracy are greatly improved, as shown in Figure 4.

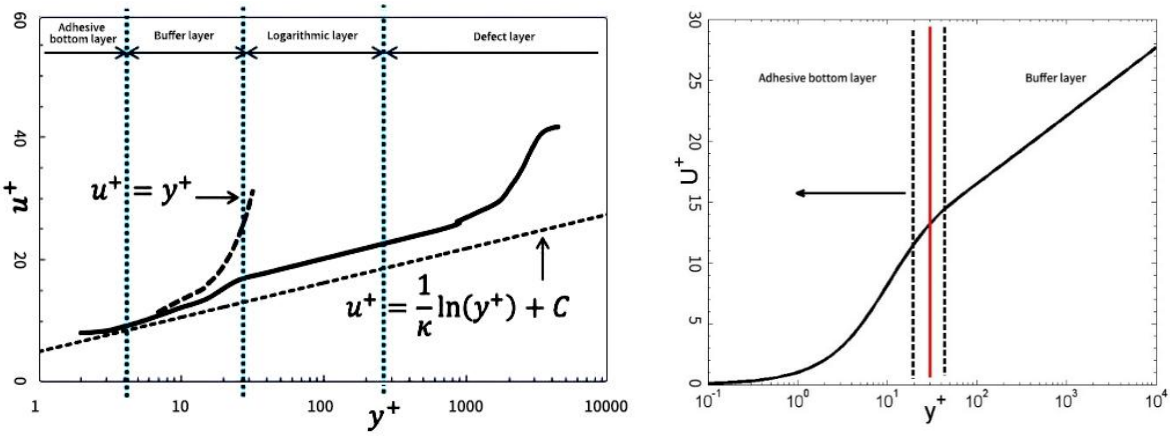


Fig 4: Wall treatment method

The final digital solution is obtained by the simulation calculation according to the internal customized iterative program, as shown in Figure 5.

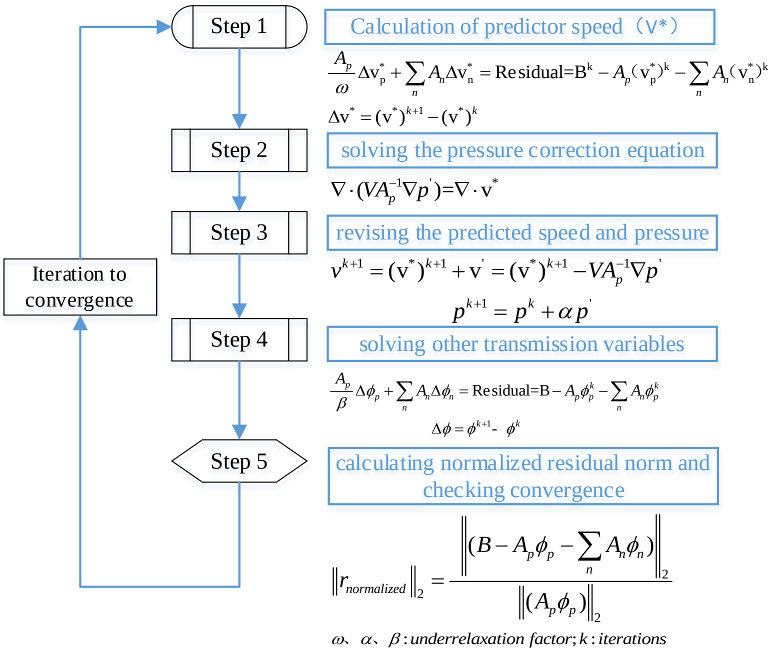


Fig 5: Calculation iteration process

III. Simulation Analysis

Based on a hydropower station under construction, the ventilation situation of the complex underground traffic tunnel group is simulated and analyzed. Firstly, through the simulation results, the layout interval of fans in tunnel group is analyzed; then, by simulating the design form of the intersection corner between the main tunnel and the branch tunnel, the influence of the fan arrangement on the airflow velocity and the influence of the fan spatial position arrangement in the branch tunnel are obtained; finally, under the above setting conditions, the airflow situation of the whole underground complex traffic tunnel group is simulated and analyzed, and the overall ventilation effect of the traffic tunnel group is obtained.

In the process of simulation calculation, if simulated according to the actual situation, not only the calculation amount is large, but also the model grid processing is relatively complex, so the calculation model is simplified according to the actual engineering size, ignoring the minor factors such as maintenance roads and drainage ditches [17]. The calculation model is based on the BIM model created by 3D Experience.

To facilitate the calculation, the gas in the tunnel will be properly assumed without affecting the accuracy of the calculation results. In practical engineering, the viscosity of gas is often ignored, and gas is regarded as ideal gas [18-20]. Therefore, the following assumptions are usually made for the gas in the tunnel:

The fluid in the tunnel is stable turbulence; 1) The fluid in the tunnel is a continuous medium; 2) The fluid in the tunnel is incompressible; 3) The fluid temperature in the tunnel is unchanged; 4) the tunnel wall is insulated; 5) the fluid in the tunnel meets the basic conservation principle; 6) the influence of tunnel wall roughness on airflow is constant.

3.1 Design and calculation of fan initial parameters and fan installation profile

FMK(Fluid Dynamics Engineer) is a CFD computing tool developed by Dassault, France based on 3D Experience

3.1.1 Calculation of air volume required for underground complex traffic tunnels

1) Air volume required for dilution of smoke

$$Q_{VI} = \frac{1}{3.6\times10^{6}} \times q_{VI} \times f_{a(VI)} \times f_{d} \times f_{h(VI)} \times f_{iv(VI)} \times L \times \sum_{m=1}^{n_D}(N_m \times f_{m(VI)}) \tag{1}$$

Wherein:Q_{VI} ——Tunnel smoke emission (m^2/s);
q_{VI} ——Standard emission of smoke [m^2/(veh ·km)];
$f_{a(VI)}$——Vehicle condition coefficient considering smoke;
f_d ——Vehicle density coefficient;
$f_{h(VI)}$ ——Altitude high speed coefficient considering smoke;
$f_{iv(VI)}$——longitudinal slope-speed coefficient considering smoke;
L ——Tunnel length (m);
$f_{m(VI)}$——Diesel vehicle model coefficient considering smoke;
n_D ——Number of diesel vehicle types;
N_M ——Traffic volume of corresponding vehicle type (veh/h).

$$Q_{req(VI)} = \frac{Q_{VI}}{K} \tag{2}$$

Wherein:$Q_{req(VI)}$——Air required for dilution of smoke in tunnel (m^3/s);
Q_{VI} ——Tunnel smoke emission (m^2/s)
K ——Design concentration of smoke (m^{-1})

2) Air required for dilution of CO

$$Q_{CO} = \frac{1}{3.6\times10^{6}} \times q_{CO} \times f_a \times f_d \times f_h \times f_{iv} \times L \times \sum_{m=1}^{n}(N_m \times f_m) \tag{3}$$

Wherein:Q_{CO}——Tunnel CO emissions (m^3/s);
q_{CO}——CO baseline emission in the design target year[m^3/(veh · km)];
f_a ——Vehicle condition coefficient considering CO;
f_d ——Vehicle density coefficient;
f_h ——Elevation coefficient considering CO;
f_{iv} ——Longitudinal slope-speed coefficient considering smoke;
f_m ——Diesel vehicle model coefficient considering smoke;
n ——Number of vehicle types;
N_m ——Traffic volume of corresponding vehicle type (veh/h).

$$Q_{req(CO)} = \frac{Q_{co}}{\delta} \times \frac{P_o}{P} \times \frac{T}{T_o} \times 10^{6} \tag{4}$$

Wherein: $Q_{req(CO)}$——air required for dilution of CO in tunnel (m^3/s);
Q_{CO}——tunnel CO emission (m^3/s);
δ ——CO concentration;
P_o ——Normal atmosphere (kN/m^2), taking 101.325kN/m^2;
P ——Tunnel site atmospheric pressure (kN/m^2);
T_o ——Standard temperature (K), taking 273K;
T ——Summer temperature at tunnel site (K).

3) Air required for ventilation in tunnel

Calculate according to the minimum ventilation frequency:

$$Q_{req(ac)} = \frac{A_r \times L \times n_s}{3600} \tag{5}$$

Wherein: $Q_{req(ac)}$——Air required for ventilation in tunnel (m^3/s);
A_r——Clearance cross-sectional area of tunnel (m^2);
n_s——Frequency of minimum ventilation in tunnel.

Calculate according to the minimum velocity of ventilation airflow in tunnel

$$Q_{rep(ac)} = V_{ac} \times A_R \tag{6}$$

Wherein: $Q_{req(ac)}$——Air required for ventilation in tunnel (m^3/s);
A_r——Clearance cross-sectional area of tunnel (m^2);
V_{ac}——Minimum velocity of ventilation airflow in tunnel (m/s).

4) Air required for fire smoke exhaust

$$\boldsymbol{Q_{rep(f)} = V_c \times A_r} \tag{7}$$

Wherein: $Q_{req(ac)}$——Air required for ventilation in tunnel (m^3/s);

A_r——Clearance cross-sectional area of tunnel (m^2);

V_c——Critical airflow velocity in tunnel fire (m/s).

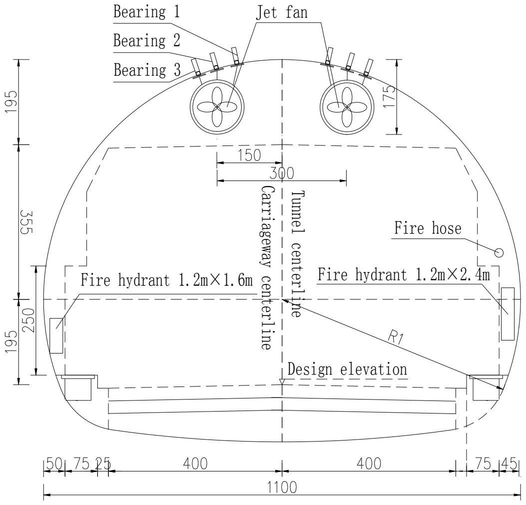
3.1.2 Configuration of jet fans in underground complex traffic tunnels

According to the traffic volume of underground complex traffic tunnels, vehicle composition and code requirements, it is preliminarily determined that jet fans are used for longitudinal ventilation in main underground complex traffic tunnels, and natural ventilation is used in short underground complex traffic tunnels. Combined with the traffic volume and longitudinal slope of underground complex traffic tunnels, and considering the road capacity, the air required for dilution of CO and smoke, ventilation and fire smoke extraction in underground complex traffic tunnels under normal operation conditions is calculated, and the maximum calculated air volume and design air volume of underground complex traffic tunnels under various conditions are comprehensively determined.

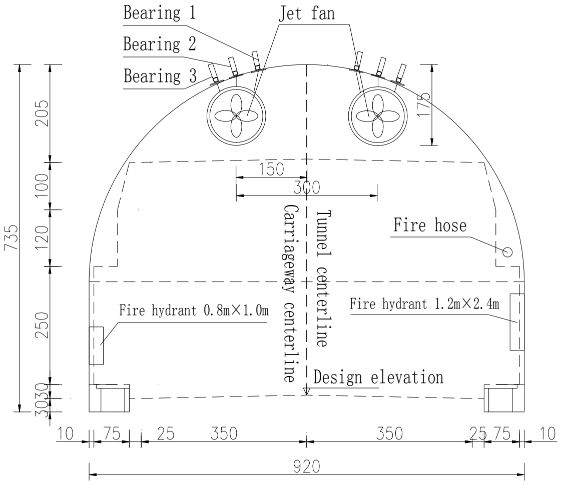
Jet fan has been widely used in longitudinal ventilation of complex underground traffic tunnels because of its small size, obvious boosting effect and convenient layout. According to the design ventilation volume of complex traffic tunnels in different places, the parameters and quantity of jet fans are determined. Parameters of jet fan: impeller diameter φ=1120 mm; outlet airflow velocity Vj=32.8 m/s; flow rate Qj=33.3 m^3/s; static thrust=1150 n; thrust/rate=36.0 n/kw; pressure drop=1000 Pa.

3.1.3 Installation position of jet fans in underground complex traffic tunnels

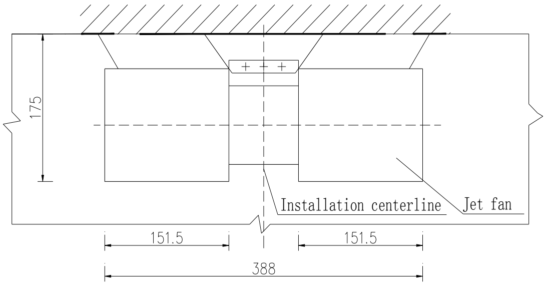
The section of fan installation position is shown in Figure 6.



Installation profile of jet fan in tunnel with 8-meter road width



Installation profile of jet fan in tunnel with 7-meter road width



Schematic diagram of jet fan profile

Fig 6: Installation section of jet fan in underground complex traffic tunnel

3.2 Strategy of ventilation and smoke exhaust control in underground complex traffic tunnels

Limited by clearance, when there is tunnel fire, the flame extends horizontally, and the hot gas can spread far along the wind. When there is a large longitudinal wind flow in the tunnel, the whole section of the tunnel will be filled with smoke, causing people to lose their way and possibly die of poisoning. Tunnel ventilation should avoid or minimize the diffusion of high-temperature smoke in the fire field, so as to prevent the hot gas from igniting vehicles outside the fire field and expanding the fire field. Therefore, tunnel fire conditions and countermeasures are the focus of ventilation design, and ventilation operation procedures must be conducive to evacuation and escape, and help firefighters approach the fire site from the windward direction to carry out fire fighting operations.

3.2.1 Main purpose of ventilation control under fire conditions

1) In the evacuation stage, it is mainly considered to be beneficial to the evacuation of personnel and vehicles, and to avoid the smoke from accident tunnel invading crosswalks, vehicular crosswalks, safe passages, adjacent tunnels and shelters, etc. The fan opening position should not be too close to the fire source, and the fan downstream of the fire source should be turned on as much as possible; 2) after evacuation, it comes to the fire extinguishing stage, which is mainly to control smoke diffusion and provide conditions for rescue work; 3) the spread of smoke can be effectively controlled and high-temperature gas can be prevented from igniting other vehicles and facilities outside the fire source to cause the spread of fire; 4) in the post-fire stage, it is mainly to discharge the smoke in the tunnel as soon as possible to provide conditions for post-disaster detection and traffic restoration as soon as possible.

3.2.2 Personnel evacuation under fire

1) Under the condition that one-way traffic is not blocked, the longitudinal airflow upstream of the fire source along the driving direction should be maintained at critical airflow velocity to ensure that smoke does not flow back; 2)

under the condition of one-way traffic blockage, low airflow velocity (e.g. 1.2 0.2m/s) should be maintained to reduce smoke backflow to the upstream of the fire source in the direction of traffic flow, and the stratification of smoke should be allowed to dilute toxic gases and ensure personnel escape; 3) under two-way traffic conditions, the airflow should be maintained at low velocity to avoid smoke backflow, unless there are other decisions (such as getting close the entrance, etc.), and smoke stratification should be maintained to ensure personnel evacuation in both directions.

3.2.3 Smoke exhaust control under fire conditions

1) Under the condition that one-way traffic is not blocked, the velocity of longitudinal airflow upstream of the fire source along the driving direction should reach the critical airflow velocity to ensure that smoke does not flow back; 2) under the two-way traffic condition, the zero airflow velocity in the exit area is ensured by controlling the airflow velocity at both ends of the exit area. In this case, the goal of airflow is to keep the smoke stratified and control the smoke in the exit area; 3) for the tunnel escape (special tunnel escape or transverse tunnel), it is necessary to maintain a positive pressure of 30-50pa and an airflow velocity of 1m/s (at the opening) in the accident tunnel.

3.3 Simulation of fan arrangement interval in the main tunnel

To understand the airflow velocity distribution in the tunnel when the fans are running, two jet fans are arranged side by side at a distance of 200m, and the actual fan parameters and relevant boundary conditions are input. The cross section of the main tunnel is shown in Figure 6. The two jet fans are arranged side by side in the tunnel, with a flow rate of 31.1m/s and a pressure drop of 1000Pa. The simulation results are shown in Figure 7. It can be seen from the figure that the airflow velocity at the outlet of the fan is relatively large, reaching 12m/s at the maximum. With the increase of the distance from the outlet of the fan, the airflow velocity gradually decays. Until the next fan is arranged, the airflow velocity is improved again, and so on. Therefore, in the main tunnel, when the fan arrangement interval is 200 meters, the average airflow velocity in the tunnel reaches about 4m/s, which meets the specification requirements and is beneficial to entrain the waste gas in the tunnel.

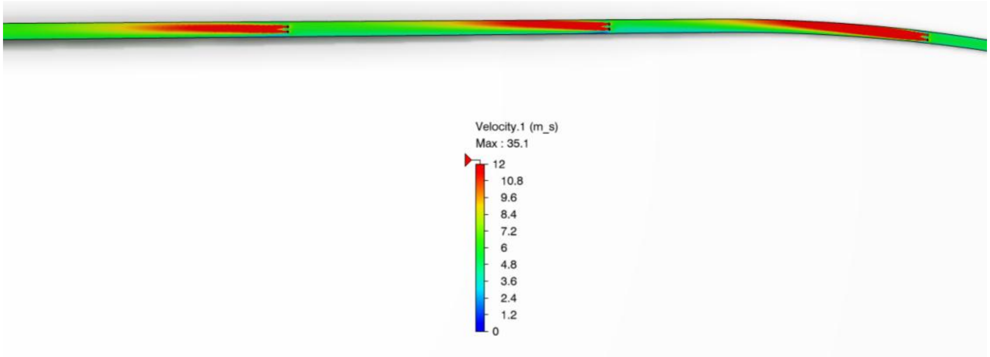
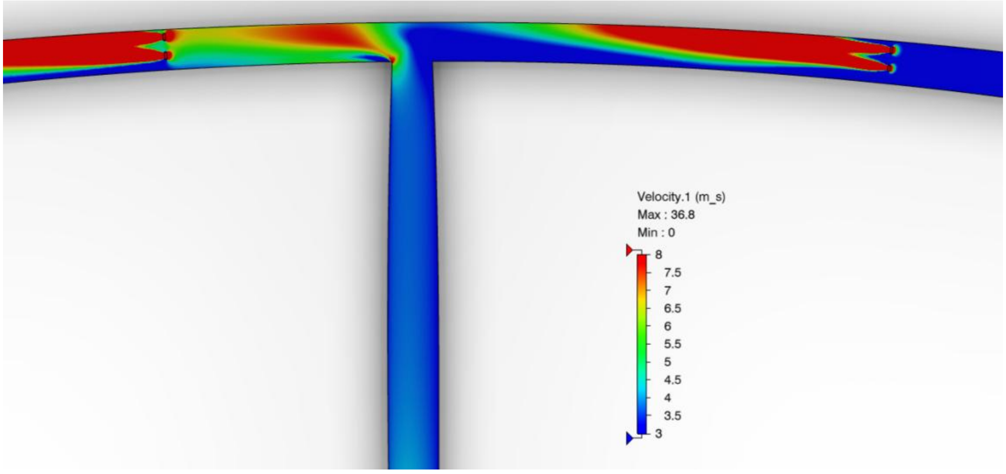


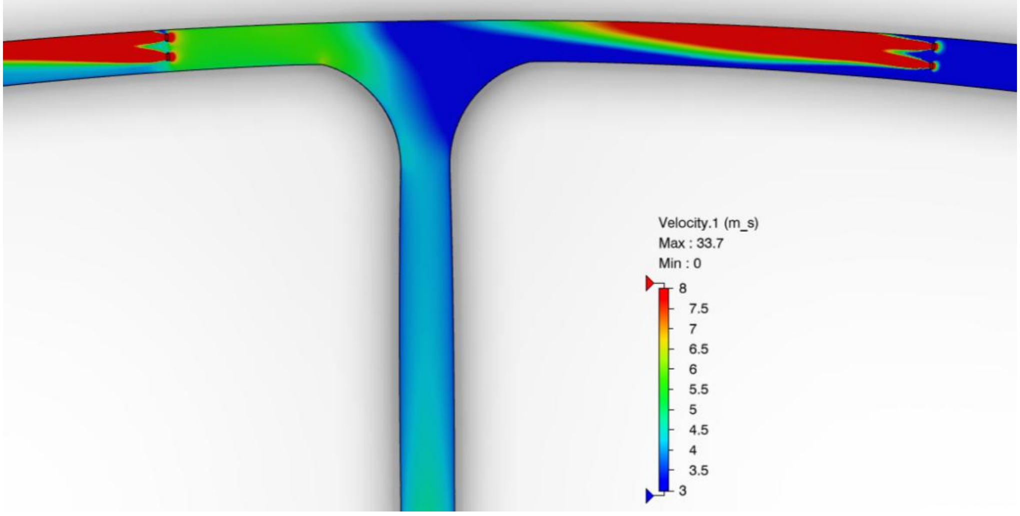
Fig 7: Distribution of airflow velocity in tunnel

3.4 The influence of the design form of the intersection between the main tunnel and the branch tunnel on the airflow velocity

The branch tunnel is mainly used as the exhaust outlet for the underground complex traffic tunnel group, and the jet fan directs the airflow in the main tunnel towards the branch tunnel. The engineering design form of the intersection of the main tunnel and the branch tunnel will affect the airflow distribution, thus affecting the ventilation effect. Therefore, this simulation model simulates the airflow distribution under two engineering design forms: right angle and chamfer angle. The simulation results are shown in Figure 8. From the nephogram distribution of airflow velocity in the tunnel obtained by simulation, it can be seen that the average airflow velocity in the branch tunnel (Figure 8(a)) is lower than that in the form of chamfering design (Figure 8(b)) when the intersection of the main tunnel and the branch tunnel is designed at right angle. Therefore, the chamfering design at the corner between the main tunnel and the branch tunnel will help to improve airflow velocity and form better airflow organization.



8 (a)

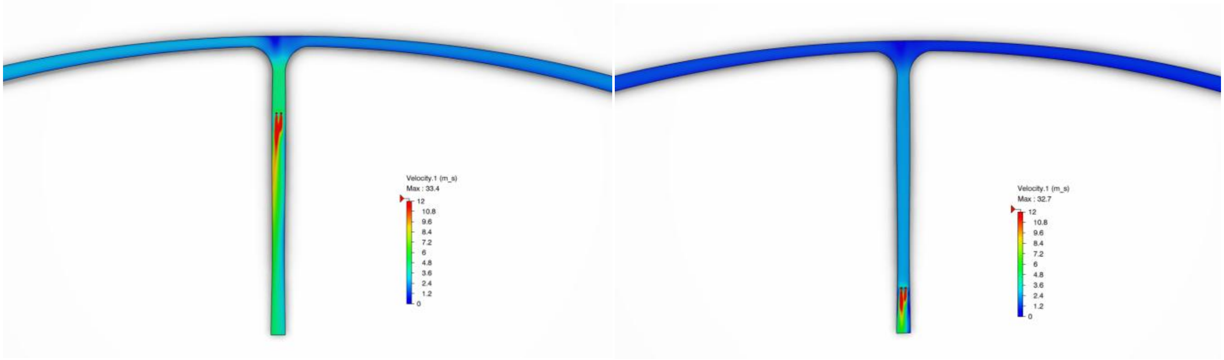


8 (b)

Fig 8: Distribution of airflow velocity in tunnel

3.5 Selection of fan arrangement position in branch tunnel

The air distribution in the branch tunnel also has a great influence on the overall exhaust effect, and different fan positions will cause different air distribution in the branch tunnel. This simulation simulates two situations in which fans are arranged at the head and end of branch tunnel respectively. The nephogram distribution of air flow velocity when fans are arranged at the head and end of branch tunnel is shown in Figure 9(a) and Figure 9(b) respectively. It can be seen from the two air flow distributions that if fans are arranged at the head near the main tunnel side in branch tunnel, larger air flow velocity will be obtained in branch tunnel, which will improve the air flow velocity in branch tunnel more significantly. Therefore, the fan in the branch tunnel should be arranged as close as possible to the side of the main tunnel, which is more conducive to improving the exhaust effect.



9 (a) *9 (b)*

Fig 9: Distribution of airflow velocity in tunnel

IV. Case Application

According to the above conclusions of fan arrangement interval, tunnel corner engineering design and fan position setting in branch tunnel, ventilation simulation is carried out for two different tunnel groups of a large hydropower station in Qinghai-Tibet Plateau.

(1) Tunnel Form I

General situation: the tunnel group contains one main tunnel and three chamfered branch tunnels, and the fans in branch tunnel are installed near the main tunnel side.

The simulation parameters are as follows: steady-state calculation, K-e turbulence model; grid size: 50-1000mm; boundary layer: 10mm 5 layers; total number of grids: 452w; number of fans: 32 fans in double rows; boundary conditions of fan: flow rate 31.1m/s, pressure drop 1000Pa. The simulation results are shown in Figure 10:

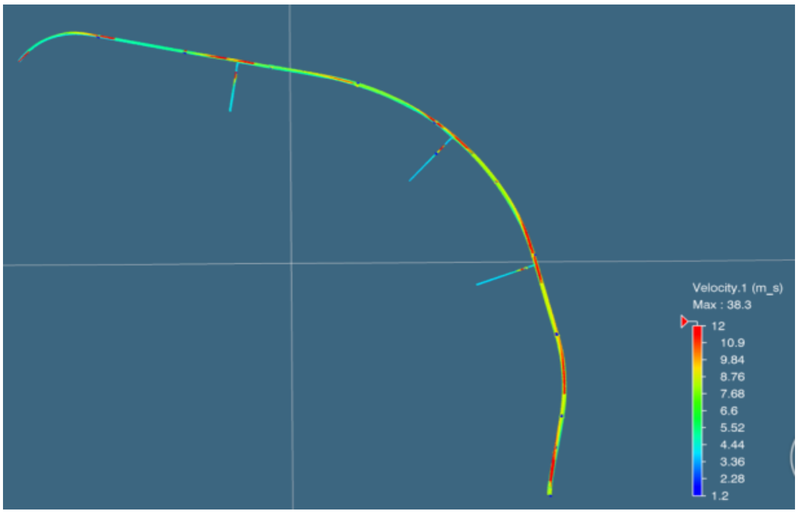


Fig 10: Distribution of airflow velocity in tunnel form I

It can be seen from the figure that, in this tunnel form, the airflow velocity in the main tunnel is above 6m/s, mainly between 6-12m/s, and the airflow velocity in the area affected by the fan and its outlet direction is relatively high.

(2) Tunnel Form II

General situation: the tunnel group includes 2 main tunnels and 5 chamfered branch tunnels, with double-row fans arranged in the main tunnel and single-row fans and double-row fans arranged in the branch tunnels, as shown in Figure 11.

The parameters are as follows: the diameter of the main tunnel: 12m; diameter of branch tunnel: 4.5m; main tunnel: 34 double-row fans in total; branch tunnel 1 to branch tunnel 3: fans in single row, with one fan per tunnel; branch tunnel 4: fans in double rows, a total of 4 fans; branch tunnel 5: fans in double rows, totally 2 fans.

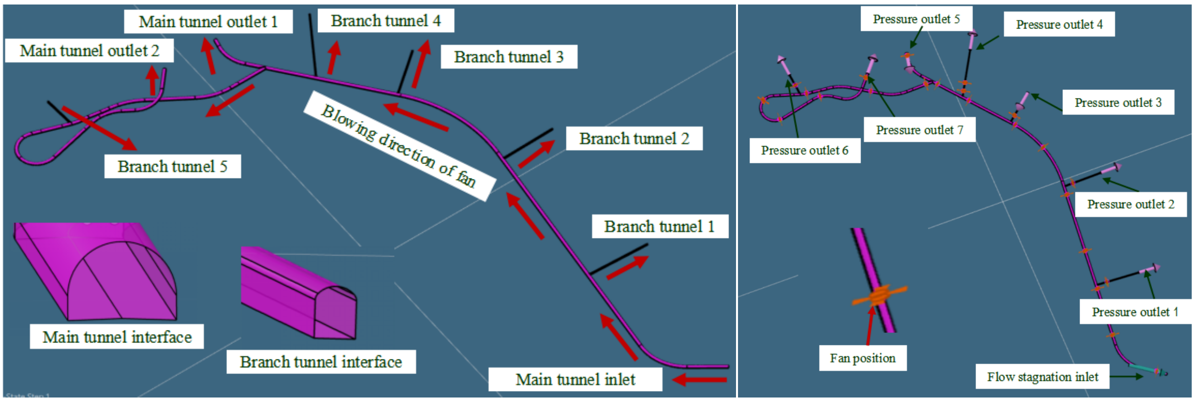


Fig 11: layout of tunnel form II

The simulation parameters are as follows: steady-state calculation and K-e turbulence model; Grid size: 50-1000mm; Boundary layer: 10mm 5 layers; Total number of grids: 452w; Number of fans: 32 fans in double rows; Boundary conditions of fan: flow rate 31.1m/s, pressure drop 1000Pa. The simulation results are shown in Figure 12.

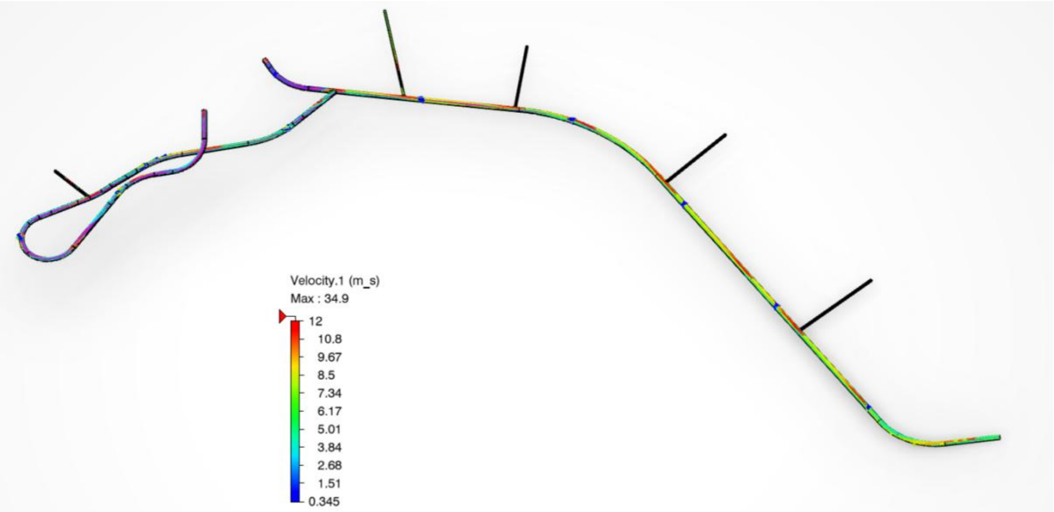


Fig 12: Distribution of airflow velocity in tunnel form II

It can be seen from Figure 12 that the main tunnel, branch tunnels 4 and 5 have obvious airflow streamline, but no obvious airflow streamline is found in branch tunnels 1-3. That is, the airflow at the entrance does not flow to branch tunnels 1-3 during the flow process. Through the analysis of fan arrangement, it can be seen that the number of fans in branch tunnels 1-3 is less than that in branch tunnels 4 and 5, and the air flow in branch tunnels 1-3 is not obvious due to the suction effect of more fans in branch tunnels 4 and 5. Therefore, when setting the number, models and specifications of fans arranged in branch tunnels, the models and numbers of fans in each branch tunnel should be kept consistent as far as possible.

Seen from the overall simulation results of the above two tunnel forms, according to the fan layout and tunnel type settings in Section 2.1-2.3 of this paper, the overall ventilation effect of the tunnel is good, with an average airflow velocity of 6-12m/s, indicating good airflow velocity. At the same time, the fan layout of each branch tunnel should be consistent as far as possible, so that effective airflow organization can be formed in each branch tunnel.

V. Summary and Prospect

In this paper, taking the complex underground traffic tunnel group of a hydropower station as a simulation model, the characteristics of mechanical ventilation with jet fans in tunnels are simulated by FMK, and the main conclusions are as follows:

(1) The layout interval of fans in the tunnel is 200m, and the average airflow velocity in the tunnel is about 4m/s, which meets the requirements of the code and is beneficial to exhaust the waste gas in the tunnel.
(2) It is advisable to round the corner at the intersection of the branch tunnel and the main tunnel, which is conducive to enhancing the airflow velocity in the tunnel.
(3) The fan in the branch tunnel should be arranged near the main tunnel side, which can improve the airflow velocity in the branch tunnel and ensure the ventilation effect.
(4) The number and model of fans arranged in each branch tunnel should be consistent as far as possible, so that each branch tunnel can form effective air distribution.

The ventilation characteristics of underground tunnels are simulated and analyzed, and the key design characteristics of the ventilation system are obtained. However, the premise of the above analysis is that the airflow in the tunnels is constant in temperature and free from external disturbance. However, the airflow will be affected by traffic flow, air temperature and external wind field during the actual operation of the tunnels. Therefore, the next research should consider the influence of external factors on the basis of theoretical assumptions, and conduct more simulation analysis of different traffic tunnels at the same time.

Acknowledgements

This paper is one of the research achievements of High-level Innovative Talents in Guizhou Province (Qiankehe Platform Talent [2018]5630-2).
This paper is supported by Guizhou Province Science and Technology Plan Project (Project Number: Qiankehe Platform Talent [2019] No.5301).

References

[1] W. Tian, X.Q. Zheng, J. Liu, et al., “Study on pollutant channeling law and new ventilation mode of extra-long tunnel group,” Modern Tunnel Technology (Supplement 1), vol. 57, pp. 629-634, 2020.
[2] C.G. Fan, J. Ji, Z.H. Gao, et al. “Experimental study on transverse smoke temperature distribution in road tunnel fires,” Tunneling and Underground Space Technology, vol. 37, pp. 89-95, 2013.
[3] F. Liu, C. Yin, J.Z. Chen, et al., “Numerical simulation study on the influence of included angle of confluence section of expressway tunnel on ventilation characteristics,” Modern Tunnel Technology, pp. 1009-6582, S1-0645-06, 2020.
[4] J. KATOLICK, M. J CHA, “Eulerian Lagrangian model for traffic dynamics and its impact on operational ventilation of road tunnels,” Journal of Wind Engineering and Industrial Aerodynamics, vol. 93, no. 1, pp. 61-77, 2005.
[5] L. FERKL, G. MEINSMA, “Finding optimal ventilation control for highway tunnels,” Tunneling and Underground Space Technology, vol. 22, no. 2, pp. 222-229, 2007.
[6] S.L. Li, Y.X. Wang, X.S. Song, et al., “Study on influencing factors of ventilation fluid velocity field in curved tunnel construction,” Chinese Journal of Underground Space and Engineering, Supplement 1, vol. 16, 2020.
[7] Y.D. Wang, S.H. Hua, Z.W. He, et al., “Hybrid ventilation design system of cross passage of extra-long

highway tunnel combined with single wind well," Journal of Traffic and Transportation Engineering, vol. 20, 2020.

[8] J. Li, X.N. Yan, X.Q. Ye, et al., "Study on the solution of full jet fire ventilation network in extra-long highway tunnel," Journal of Safety Science and Technology, vol. 17, 2021.

[9] J.G. Zheng, J.L. Zheng, B. Huang, et al., "Innovation and practice of key technologies in Erlangshan tunnel construction of Yakang Expressway," Tunnel Construction, vol. 41, no. 2, 2021.

[10] J. Kong, Z.S. Xu, J. Huang et al., "Study on the influence of tunnel slope change on longitudinal ventilation efficiency of full jet," Journal of Safety and Environment, vol. 20, 2020.

[11] M.N. Wang, T. Deng, L. Yu, "Study on theoretical prediction method of construction ventilation time based on large-section high-speed rail tunnel," Tunnel Construction, vol. 38, no. 8, pp. 1279-1285, 2018.

[12] S.X. Deng, P. Cheng, "Calculation of air pollutant concentration and ventilation volume in longitudinal ventilation tunnel," China Journal of Highway and Transport, vol, 15, no. 1, pp. 89-91, 2002.

[13] M. Deng, Q. Min, Y. Xiong, et al., "Experimental study on "complementary + smoke exhaust shaft" combined ventilation model based on similarity theory of hydrodynamics," Journal of China & Foreign Highway , vol. 41, 2021

[14] Z.R. Zhu, X.Z. Li, S. Chen, "Numerical simulation of ventilation characteristics in deep-buried long tunnel construction of water diversion project," Journal of Harbin Engineering University, vol. 40, no. 7, pp. 1304-1310, 2019.

[15] T. Ye, J.D. Zhao, S.J. Zou, et al., "Numerical simulation of piston wind in highway bifurcation tunnel," Building Energy & Environment, vol. 39, no. 11, pp. 40-43, 2020.

[16] S.M. Yang, W.Q. Tao, "heat transfer (4th edition)," Beijing: Higher Education Press, 2006.

[17] K. Hu, X. Shao, "Research on longitudinal jet ventilation system of highway tunnel based on CFD," Southern Energy Construction (Supplement 2), vol. 7, 2020.

[18] M B. SEKULARAC, "Experimental determination of tunnel ventilation axial ducted fan performance," Thermal science, vol 5, 20, no. 1, pp. 209-221, 2016.

[19] Z.M. CAO, "Study on ventilation characteristics and key technol- ogies of long tunnels and complicated underground engineering in the construction stage," Chengdu: Southwest Jiaotong University, 2016.

[20] Y.H. SUN. "Calculation and simulation of natural ventilation in qinling zhongnanshan tunnel," Xi'an: Chang'an University, 2014.